

From owner-qrp-l@netcom.com Fri Mar 17 20:16:09 1995  
Message-Id: <9503171625.AA27812@link.com>  
From: U4J96@mhsgate.bgm.link.com (Chodkowski, Steve)  
Date: Fri, 17 Mar 1995 11:08 EST  
Subject: <None>

Regarding the MFJ dsp filter: it appears this filter has gotten a thumbs up review from the one of the recent ham mag reviews(qst or cq). Has any one used this filter and can you give me a "user report" on it. I need a filter for cw and ssb, qrp and qro, the radio I use now does not have any filtering. I am in the process of building the sw cap filter that was described in QST several years ago.(the one that uses the hard to find national MF8CCN chip); but I think that if it plays up to its billings the MFJ device may fulfill all needs, including any future digital modes of operation that I may try out. Any comments? Any user reports? Please send to my email direct ( not through the list)

Tks es 73 ....stevech.....  
schodkowski@bgm.link.com

From owner-qrp-l@netcom.com Fri Mar 17 19:04:44 1995  
Subject: Can Anyone Help?  
From: brian.carling@acenet.com (Brian Carling)  
Message-Id: <2a6.15413.500@acenet.com>  
Date: Fri, 17 Mar 1995 07:42:00 -0500

>From: brian.carling@acenet.com

In Message-ID: <3160@gqrp.demon.co.uk> you wrote:

G3>I am looking into finding CW Filters for the FT707 for OK1DZD.

G3>I recall adverts in US magazines by the Fox Tango Club...

G3>Does anyone have an address?

G3>--

G3>-----

G3>George Dobbs G3RJV "It is vain to do with more,

G3>G-QRP Club what can be done with less."

G3>----- William of Occam (1290-1350)

George, I tried E-mailing you about some other issues last week, but the mail came back as "undeliverable" to that address above!

What is wrong? ANYway, it wasn't important, I was just saying hello as an ex-G (G3XLQ), but I think I have the address you need there:

International Radio & Computer Inc.  
3804 South U.S. 1 (407) 489-5609 / (305) 879-6868  
Ft. Pierce, FL 34982 FAX: (407) 464-6386

Crystal filters. Icom & Kenwood newsletters. Catalog \$1.00

I hope that is correct!

```
      \--\--/
     /  \  \
    .    /    \    .
   | \ | (o)(o) | / |
  ---.000o----oo----o000.---
 |                               |
 |   Brian Carling             |
 |   Radio AF4K                |
 |   Gaithersburg, MD          |
 |                               |
 | brian.carling@acenet.com    |
 |                               |
 |-----0ooo.-----|
   .ooo0   (   )
  (   )   ) /
   \ (   ( _/
    \_)

```

---

~ SLMR 2.1a ~ I'll never forget the time I had amnesia.

From owner-qrp-l@netcom.com Sat Mar 18 02:27:15 1995

From: PDouglas12@aol.com

Date: Fri, 17 Mar 1995 23:50:46 -0500

Message-Id: <950317235045\_53113586@aol.com>

Subject: Dayton or YKW

That's Dayton or You Know What! I am getting so psyched, I am boring everyone in my household to death. My xyl has lived through Dayton before and is philosophical about the whole thing. But I am a basket case. Anyway, for you other fanatics, here's the latest updated list of us QRP-L subscribers who will be at Dayton and for whom I am printing ID badges to help us find each other among the uninitiated. Still plenty of time to send in your call and full name by direct email to pdouglas12@aol.com. This posting should include all messages received as of Friday morning the 17th.

Any received after will be on next week. There are now over sixty of us going. An amazing number, considering the small pool of us to start with. All requested corrections have been made, so check your listing--reemail any corrections still not right. Only dumb thing I did was alphabetize by first name. Rich Richmond, don't panic, you're on the bottom till next posting!

Too late to change to last names, so that's how they'll be at Dayton unless I get ambitious and physically rearrange the badges after printing. Nah.

You'll find 'em. 72 Preston WJ2V

ANTHONY LUSCRE KA8NRC  
BILL KELSEY N8ET  
BOB GOBRICK V01DRB/WA6ERB  
BOB MARLAN KA6NOC/8  
BOB STAFFORD N9USD  
BRUCE FLORIP AA7AR/6  
BRUCE LIFTER AD4TG  
BRUCE MUSCOLINO W6TOY/3  
BUCK SWITZER N8CQA  
BYRON JOHNSON WA8LCZ  
CHUCK ADAMS K5FO  
CLARK FISHMAN WA2UNN  
DAVID FELDMAN WB0GAZ  
DAVID MEACHAM W6EMD  
DEE ANN RAY N9XLV  
DENNIS BLANCHARD K1YPP  
DICK PASCOE G0BPS  
DICK SZAKONYI KA3ZOW  
DOUG HENDRICKS KI6DS  
DUFFY BEISCHEL WB8NUT  
ERIC SWARTZ WA6HHQ  
F. "MITCH" MITCHELL WA4OSR  
GARY DIANA N2JGU  
GEORGE SILVER N9SXB  
HANK KOHL K8DD  
HOWIE CAHN WB2CPU  
JANE BLANCHARD KA1FUN  
JASON PENN N9RPT  
JERRY SY N3RKD  
JIM CATES WA6GER  
JIM FITTON W1FMR  
JIM JOHNS KA0IQT  
JIM KORTGE NU8N  
JIM STAFFORD W4QO  
JOE EVERHART N2CX  
JOHN ROUSE KA3DBN  
KATY SZAKONI N3SAD  
KEN EVANS KJ4XR  
LARRY MAKOSKI N2ELW  
LINDA LUSCRE KA8ODP  
LOWELL CORBIN KD8FR  
MARTY HARTWELL KD8BJ  
MARTY RAY NN9H  
MICHAEL RIOUX NW1J  
MIKE CZUHAJEWSKI WA8MCQ  
MIKE FLANNAGAN KB8NKX  
MONTE "RON" STARK KU7Y  
NILS YOUNG WB8IJN

PAUL VALKO WB8ZJL  
PAULETTE QUICK N9OUH  
PETE MEIER WK8S  
PETER BEEDLOW NN9K  
PRESTON DOUGLAS WJ2V  
PRESTON DOUGLAS WJ2V  
REV. GEORGE DOBBS G3RJV  
ROBIN CORBIN NI9R  
RON DOYLE N8VAR  
RON MAJEWSKI WB8RUQ  
STEVE COHEN N30IE  
STEVE HIDEG N8HSC  
TED ALBERT KF8EE  
TOM FRISZ N9DD  
TOM LIFLAND W2RFU  
TOM STAFFORD N9YBC  
WAYNE BURDICK N6KR  
RICH RICHMOND N4AFX

From owner-qrp-l@netcom.com Sat Mar 18 03:20:31 1995  
From: PDouglas12@aol.com  
Date: Fri, 17 Mar 1995 23:51:24 -0500  
Message-Id: <950317235123\_53114068@aol.com>  
Subject: Dayton or YKW

That's Dayton or You Know What! I am getting so psyched, I am boring everyone in my household to death. My xyl has lived through Daytons before and is philosophical about the whole thing. But I am a basket case. Anyway, for you other fanatics, here's the latest updated list of us QRP-L subscribers who will be at Dayton and for whom I am printing ID badges to help us find each other among the uninitiated. Still plenty of time to send in your call and full name by direct email to pdouglas12@aol.com. This posting should include all messages received as of Friday morning the 17th. Any received after will be on next week. There are now over sixty of us going. An amazing number, considering the small pool of us to start with. All requested corrections have been made, so check your listing--reemail any corrections still not right. Only dumb thing I did was alphabetize by first name. Rich Richmond, don't panic, you're on the bottom till next posting! Too late to change to last names, so that's how they'll be at Dayton unless I get ambitious and physically rearrange the badges after printing. Nah. You'll find 'em. 72 Preston WJ2V

ANTHONY LUSCRE KA8NRC  
BILL KELSEY N8ET  
BOB GOBRICK V01DRB/WA6ERB  
BOB MARLAN KA6NOC/8  
BOB STAFFORD N9USD

BRUCE FLORIP AA7AR/6  
BRUCE LIFTER AD4TG  
BRUCE MUSCOLINO W6TOY/3  
BUCK SWITZER N8CQA  
BYRON JOHNSON WA8LCZ  
CHUCK ADAMS K5FO  
CLARK FISHMAN WA2UNN  
DAVID FELDMAN WB0GAZ  
DAVID MEACHAM W6EMD  
DEE ANN RAY N9XLV  
DENNIS BLANCHARD K1YPP  
DICK PASCOE G0BPS  
DICK SZAKONYI KA3ZOW  
DOUG HENDRICKS KI6DS  
DUFFY BEISCHEL WB8NUT  
ERIC SWARTZ WA6HHQ  
F. "MITCH" MITCHELL WA4OSR  
GARY DIANA N2JGU  
GEORGE SILVER N9SXB  
HANK KOHL K8DD  
HOWIE CAHN WB2CPU  
JANE BLANCHARD KA1FUN  
JASON PENN N9RPT  
JERRY SY N3RKD  
JIM CATES WA6GER  
JIM FITTON W1FMR  
JIM JOHNS KA0IQT  
JIM KORTGE NU8N  
JIM STAFFORD W4QO  
JOE EVERHART N2CX  
JOHN ROUSE KA3DBN  
KATY SZAKONI N3SAD  
KEN EVANS KJ4XR  
LARRY MAKOSKI N2ELW  
LINDA LUSCRE KA8ODP  
LOWELL CORBIN KD8FR  
MARTY HARTWELL KD8BJ  
MARTY RAY NN9H  
MICHAEL RIOUX NW1J  
MIKE CZUHAJEWSKI WA8MCQ  
MIKE FLANNAGAN KB8NKX  
MONTE "RON" STARK KU7Y  
NILS YOUNG WB8IJN  
PAUL VALKO WB8ZJL  
PAULETTE QUICK N9OUH  
PETE MEIER WK8S  
PETER BEEDLOW NN9K  
PRESTON DOUGLAS WJ2V

PRESTON DOUGLAS WJ2V  
REV. GEORGE DOBBS G3RJV  
ROBIN CORBIN NI9R  
RON DOYLE N8VAR  
RON MAJEWSKI WB8RUQ  
STEVE COHEN N3OIE  
STEVE HIDEG N8HSC  
TED ALBERT KF8EE  
TOM FRISZ N9DD  
TOM LIFLAND W2RFU  
TOM STAFFORD N9YBC  
WAYNE BURDICK N6KR  
RICH RICHMOND N4AFX

From owner-qrp-l@netcom.com Fri Mar 17 21:40:07 1995  
From: Mike.Czuhajewski@hambbs.wb3ffv.ampr.org (Mike Czuhajewski)  
Subject: Ed Hare KA1CV at Timonium, MD  
Date: Fri, 17 Mar 95 19:05:40 EST5EDT  
Message-Id: <1995Mar17.190540.19121@wb3ffv.ampr.org>

I don't know if he even lurks in the shadows of qrp-l, but Ed Hare, KA1CV, is a QRPer on the ARRL lab staff. W6TOY (Bruce3900@delphi.com) told me that Ed will be speaking at the Timonium, MD hamfest on some non-QRP topic, and will be staying in the area for a bit.

He "belongs" to W3ABC, the Division Director, for most of the time, but does have some free time, too. I forget when Bruce said Ed would be available, but think it will Friday night, which is a week from today, on 24 March. We are planning on doing what we did with N00CT when he came to Rockville, MD--meet up with Ed at his hotel (Days Inn in Timonium), and then go out to some local restaurant for dinner and QRP talking.

Details will follow, but any QRPers in the area who want to join us are more than welcome. Time will probably be something like 1900 hours at the hotel; details to follow. Timonium is just outside the Baltimore Beltway on the north side, on Route 83 (which goes up to York, PA). We do ask that you notify me or Bruce so we'll know whether we should wait for anyone else.

73 and Queue Our Pea DE WA8MCQ

--

Mike Czuhajewski, user of the UniBoard System @ wb3ffv.ampr.org  
E-Mail: Mike.Czuhajewski@hambbs.wb3ffv.ampr.org  
The WB3FFV Amateur Radio BBS - Located in Baltimore, Maryland USA  
Supporting the Amateur Radio Hobby, and TCP/IP InterNetworking

From owner-qrp-1@netcom.com Fri Mar 17 18:56:47 1995  
From: jpo@acd4.acd.com ( Jim Osburn )  
Message-Id: <9503172023.AA23903@IEDV5.acd.com>  
Subject: FAR At The Fest  
Date: Fri, 17 Mar 95 15:23:13 EST

Last Sunday at the Indiana Hamfest I purchased a Curtis Keyer board from FAR Circuits. The chip and socket were supposed to be included but they were out of stock. I was told the chip and socket would be shipped as soon as they came in, no shipping charge. Today the chip and socket arrived.

I feel good about buying stuff from FAR Circuits at hamfests. There's a nice big display of all the circuit boards to choose from. I can inspect the board before I buy and I don't have to pay shipping.

FAR Circuits was also selling scrap unetched PCB material at the fest. The pieces were about 4 by 6 and were 20 cents a piece, or six pieces for a \$1.00. I bought six pieces.

73,

Jim, WD9EYB

From owner-qrp-1@netcom.com Fri Mar 17 19:02:28 1995  
Message-Id: <n1416669689.60732@msmailgw1.arlut.utexas.edu>  
Date: 17 Mar 1995 15:04:25 -0600  
From: "rohre" <rohre@msmailgw1.arlut.utexas.edu>  
Subject: GQRP/George Dobb's messages

My system reports a problem with the demon server in U.K. accepting George's mail, it will automatically retry for 5 days, so will see if it eventually works to his regular address he used in his recent query. The Yaesu crystal filters are advertised in the back of 73, in the classifieds, from the company in Ft. Pierce FL.

From owner-qrp-1@netcom.com Sat Mar 18 02:32:45 1995  
From: Jerry Palsson <palsson@rohan.sdsu.edu>  
Message-Id: <199503180430.UAA07401@rohan.sdsu.edu>  
Subject: Re: How about a WWW site?  
Date: Fri, 17 Mar 1995 20:30:22 -0800 (PST)

Bob,

Sure--a Web server not only provides ftp links but also gopher and telnet and in some cases, e-mail, news, etc. as well. It provides a kind of all-in-one connectivity that the Internet has sorely needed for a long time.

That's the beauty of it.

But my point was simply that a server devoted primarily to images could disappoint people if they expect it to be efficient. The best way to view an image is still the local way: have it residing on your own system.

That said, I hope I didn't sound discouraging. I still think a Web site for QRPers would fill a void. I'd love to see one set up. It could provide a lot more than just pretty pictures, of course.

>So even though there may be great ftp sites like yours that store images etc  
>- the WWW homepage can act as the "one-stop'shopping" place to get there  
>(ans other qrp ftp related files).

Sorry, you misunderstood. I didn't say I have an ftp site; I mentioned only my Web server. (But just for the record: I also have an ftp server, yes.)

Thanks for the response.

Jerry AA6KI  
email: palsson@bookworm  
<http://bookworm.sdsu.edu/>

From owner-qrp-1@netcom.com Sat Mar 18 02:43:24 1995  
From: Jerry Palsson <palsson@rohan.sdsu.edu>  
Message-Id: <199503180416.UAA05708@rohan.sdsu.edu>  
Subject: Re: How about a WWW site?  
Date: Fri, 17 Mar 1995 20:15:59 -0800 (PST)

Right--Lynx \*is\* an excellent text-based Web browser and should be available to most people with UNIX accounts and TCP/IP connectivity. So a Web site for this list would be accessible to them even if they don't have Netscape or one of the many other GUI browsers.

It appears that many people here would like to see a QRP Web site established. I would encourage Steve H. or anyone else interested to go for it!

Jerry AA6KI

From owner-qrp-1@netcom.com Sat Mar 18 03:16:17 1995  
Date: Fri, 17 Mar 95 20:26:51 PST  
Message-Id: <9503180426.AB00369@altair.csustan.edu>  
From: dh@altair.csustan.edu (Doug Hendricks)  
Subject: Joining NorCal

I have received several messages on how to join NorCal and get copies of QRPP.



QRPP is the journal of the NorCal QRP Club and is published 4 times per year in March, June, September and December. It costs \$10 per year for US members, \$15 (US Funds) for Canadian members, and \$20 (US Funds) for DX members. The added cost for Canadian and DX members is for postage. It is sent airmail to those members. US members get their's sent Bulk Rate. It doesn't cost anything to join NorCal, but you won't get the journal if you don't subscribe. QRPP is 72 pages in a 5.5" x 8.5" format and uses 9 point Times New Roman type. It is jammed with articles of all types, and has NO advertising, just pure meat. To subscribe: Send a check or money order made out to Jim Cates, NOT NorCal to:

Jim Cates, WA6GER  
3241 Eastwood Rd.  
Sacramento, CA 95821

Jim and I will both be at Dayton and are looking forward to meeting all of you.  
72, Doug  
KI6DS

From owner-qrp-l@netcom.com Fri Mar 17 19:17:08 1995  
Date: Fri, 17 Mar 1995 14:31:15 -0500  
Message-Id: <199503171931.AA12247@yfn.ysu.edu>  
From: ak238@yfn.ysu.edu (Keith M. Hamilton)  
Subject: MFJ-1621 (AGAIN!)

Thanks to all who wrote concerning my question about the portable antenna MFJ-1621. I will attempt to give you some of the notes I was given.

To all who wrote asking me to e-mail them any results I get from anyone..... I will be getting all my answers here on the list so you will be able to read them right here.

To all who wrote suggesting alternatives..... I can not put up ANY type of permanent antenna! When I am done operating it must come down..... I am NOT working from my home or apartment so I can't use any attic or hidden antennas..... I DO intend to use the antenna outdoors and AWAY from any electrical interference or other signal absorbing material.

If anyone has actually used this antenna with either good or bad results please post your comments to the list!

Thank you.

--

Keith M. Hamilton - N08Z  
ak238@yfn.ysu.edu  
Youngstown, OH

Piano Technicians Guild  
Youngstown Chapter

From owner-qrp-1@netcom.com Sat Mar 18 00:41:22 1995  
Message-Id: <2f6a4827.pandora@pandora.lugs.po.my>  
Date: Sat, 18 Mar 1995 10:40:37 +0800  
From: "W. Daniel, 9V1ZV" <daniel@pandora.lugs.po.my>  
Subject: PC Boards

## Fabricating Printed Circuit Boards

### Introduction

Since the beginning of my electronics construction hobby, I've just about built circuits in every possible way I could think of. I still remember my early days when I actually had germanium transistors screwed down on a block of wood. That was before I acquired my first soldering iron. I then progressed to the well-known "ugly" construction technique where component leads were simply soldered together. Other methods I remember include, breadboards, matrix boards which were quite popular at one time, wire-wrapping, board-excavation and printed circuit boards. Of all these, I have found the printed circuit board method to be one of the most satisfying and aesthetically pleasing, so far. This does not mean that it was an easy method, in fact, I had avoided circuits requiring printed circuit boards until I had tried everything else, and there was a good reason for this. There are also limits to what you can produce yourself, for example, you will not have the facility to fabricate through-plated boards or multi-layer boards. For these, if you ever need them, its best to go to the PCB manufacturer to have them made for you.

Like many, I had supposed, rightly, that the fabrication of printed circuit boards was a tedious process, requiring a lot of basic tools and equipment which I did not possess. As such, the whole process of fabricating a printed circuit board had remained pretty much a mystery to me until I entered the field of radio electronics. I discovered then that the parasitic capacitance and electrical properties of other construction methods were simply unsatisfactory for circuits operating in the RF regions. Thus began my quest for the printed circuit board. The purpose of this article is to briefly outline the various methods and steps of producing a usable printed circuit board.

## Basic Theory

The whole idea of making a printed circuit board is really a chemical process of removing copper from the circuit board at the right places, in order to leave behind tracks suitable for carrying current as required by the circuit in question. Boards can be bought which come with copper cladding one or both surfaces, used for making single or double sided PCBs respectively. A visual survey of the board reveals an uninteresting copper surface with no holes or tracks. The constructor must find some way of removing the copper in order to obtain a circuit pattern which can be used. One way to do this is by "excavating" the board with a sharp cutting tool to remove the copper. This proves to be an extremely tedious method and can only be used for the simplest circuit patterns. It is sometimes useful for making quick and dirty modifications to an existing pattern.

The standard way of creating a pattern, however, is by means of a method known as "etching". Etching is a chemical process whereby the unwanted copper is removed by a process of reduction, a chemical reaction, which "corrodes" away the copper. This is often achieved by immersing the copper clad board into a solution of Ferric Chloride or Sodium Persulphate (I think). Upon contact, a reaction occurs which reduces the copper on the PCB to copper chloride which comes off the board. Ferric Chloride has now become a controlled substance in some places because of its toxic and environmentally unfriendly nature and this makes it difficult to obtain from the usual sources. Sodium Persulphate is the substitute for Ferric Chloride but has not arrived at the shops here yet. Either of the etchants can be obtained in liquid or in crystallized form. Nowadays, they are rarely found in liquid form because of the high rate of oxidation which renders the chemical useless. More often than not, they come in the form of dehydrated crystals which we mix with water to produce the etching solution. This solution is then used to etch the boards.

If you have been asking yourself how the chemical can be prevented from removing all of the copper, then you have asked an appropriate question. There are a number of methods by which this can be done. The whole principle lies in preventing the etching solution from coming into contact with the copper you wish preserved on the board. This can be by means of a water-proof tape on the board, rub-on transfers, marker ink, etch-resist ink, toner ink, plotter ink or photo-resist. In short, any method that can protect the board from the chemical solution is usable. Some methods are easier than others, of course, and the method to use depends on a number of factors, such as complexity of the pattern you want to produce, the density of the tracks, the boards available

etc.

Once you have, for example, drawn a pattern onto the clean copper clad board with etch-resist ink, you may immerse the board into a properly prepared etching solution for the etching process. This can take from 5 to 30 minutes or more, depending on the concentration of the etching solution. After that, the board is removed from the solution, revealing a board where the unmarked portions of the board has no copper on it, the board material beneath being now visible, and the marked parts of the board still having the ink in it. Once the ink is removed either by sanding or by the use of solvents, the copper will become visible. The one last remaining step is to drill the holes for component leads in the right places and the board is ready for use.

That is generally what happens in the process of fabricating a PCB. Time does not permit me to cover all the methods that you can use so I will highlight some of the major steps in the following text.

### The Copper Clad Board

Boards suitable for etching can easily be obtained from parts suppliers and come in a number of varieties. Typically, there are two major types of board materials that are used for the base board, fibre-glass (glass-epoxy) and phenolic paper. Fibre-glass boards tend to be tougher, look better and probably has slightly lower surface capacitance properties, as well as being the more expensive of the two. Phenolic paper, on the other hand, is easier to cut and drill though it tends to crack or fragment as it is more brittle, and cheaper. Both types can be used for homebrew construction projects as the mentioned RF properties are quite insignificant until VHF frequencies and above.

You will find single-sided copper clad boards as well as double sided ones and you choose the type of board appropriate for the circuit board you wish to fabricate. Double sided boards are normally used for RF related circuits because it offers a stable one-point ground-plane which helps stability and prevents unwanted oscillations or ground loops. However, producing double-sided PCBs requires high precision tools and I will only briefly mention some of the methods for producing simple ground-plane double sided boards later. This should be sufficient for a start.

You will also come across boards which are "pre-sensitized" or "photo-sensitized" which are used to produce photo-resist patterns. These boards are typically much more expensive than the plain boards because of the photo-resist film that has been

pre-deposited over the surface. Such boards often come in light-proof wrapping with an additional layer of opaque plastic on the board surface which will not be removed until ready for exposure. I will detail this technique below.

Before you get started with the boards however, you will need to cut the board down to the size you need. Normally boards are sold in several sizes so you may pick either a large sized board and cut it up as you need, or select a size that is closely matched with your required size to minimize the cutting. The boards can be difficult or easy to cut, depending on the material of the board and its thickness. Average boards are about 2 mm thick and are quite tough and difficult to cut. Some boards come in 1 mm thickness and can be cut using a heavy-duty cutter. This type of board is usually quite flexible and thus the bending will not damage it. The thicker types on the other hand, will tolerate little bending before fracturing or fragmenting. Cutting can be achieved using a hobby saw with a fine-serrated blade. Thick saw blades are not suitable for this. Sawing should be done slowly and gradually in order not to damage the board. Saw perpendicularly to the board, which should be clamped down to the work-bench firmly using a G-clamp or a similar device. Alternatively, you can use a heavy-duty Exacto knife to engrave the border lines of the appropriate size. When using this method, it is not necessary to completely cut through the board. Once about  $3/4$  of the thickness has been cut, you can usually snap the board along the engraved lines. It is important to engrave BOTH sides of the board, otherwise you will not get a clean break. This method is very tedious, time consuming and tends to destroy your blade, especially when cutting fibre-glass boards. It can be used when a hobby saw is not available. Try not to scratch or damage the copper surface when doing this. After the board has been cut down to size, use a medium sized file to smooth out the edges for a nice finish.

## The Etching Process

The next most important component you require is the etching solution. As of now, very few shops will sell Ferric Chloride crystals to unlicensed buyers and Sodium Persulphate is not publicly available. There is a good reason for this but it means a lot of inconvenience for the home constructor. There are means of obtaining the chemicals which will not be detailed here.

Ferric Chloride is most commonly available as dehydrated crystals and sold in plastic containers. It is very important to keep these crystals in a dehumidified environment as it tends to combine with moisture in the atmosphere and turn into a really messy and staining liquid. Be forewarned that this

substance stains permanently on clothes and even some plastics or ceramics, is highly corrosive, carcinogenic and toxic. As such it should be kept out of reach from children and water. For the same reasons, it should not be discarded into the public drainage system before diluting it with large amounts of water.

Sodium Persulphate is a white crystal and though it is environmentally more friendly than Ferric Chloride is, similar precautions should be taken and care exercised when dealing with concentrated chemicals of any type. This substance is considerably safer however. For one, it is endothermic when dissolved in water and the resultant solution is a clear and non-staining solution. It is also slower acting than Ferric Chloride and probably needs more agitation and perhaps a little warming up. A good way to speed up the reaction may be to dissolve the Sodium Persulphate crystals in boiling water. Take all necessary precautions to avoid scalding.

The way to prepare the solution is to mix the crystals into some water, usually 1 part crystals to 5 parts water. This is just a guide and once you understand the process you can easily produce higher concentrations to etch boards more quickly. You should also be aware that the process of hydrating these crystals is a highly exothermic one so do not be surprised if the water starts to boil. As such, one should NEVER throw any substantial amount of crystals into the water. Similarly, one should NEVER add water to crystals, always crystals to water.

Normally, a plastic tray suitable for immersing the circuit board is filled with about 2 cm of water. The crystals are then added to the water BEFORE putting in the board, using a plastic spatula or any other suitable instrument. The instrument MUST be dry before applying to the crystals. Never leave the crystals exposed to atmospheric air for long. As soon as you have taken out enough crystals, wipe dry the rim of the crystal container and re-seal it in its air-tight container and store in a dry place out of reach of children. Do not get the crystals or solution on to your skin or eyes, and if you do, rinse under cold running water to remove it. See a doctor immediately in the event of ingestion. As you add the crystals to the water, the water will change color, to dark brown if using Ferric Chloride, and you should notice some heat being produced. Do not be too worried by the heat as it is useful for the etching process. Do not inhale any fumes produced during the entire process, these are poisonous and though in very small amounts, may cause asphyxiation (Chlorine). All this should be done after you have readied the board for etching. All instruments coming into contact with the solution should be non-metallic. Stir the solution until all the crystals have dissolved to produce an evenly colored solution. Now the solution is ready for use. Try to

use it while it is hot so this step should always be done after your board is ready.

Put your resist-masked board into the solution slowly so as not to cause a splash. Remember that the solution is very hot sometimes. Once the board is completely immersed, regularly agitate the tray and pay attention to the exposed copper. After sometime, the exposed surface will appear dull, not necessarily evenly. The after more agitation you will see patches of circuit board becoming exposed. Do this until ALL the unwanted exposed copper surface has been removed and the board material is visible beneath it. This may not be easy initially as the etching solution may obscure your view of the board. It is therefore good to have a deeper tray which allows you to tilt the tray to expose the board. Normally, surfaces with less exposed copper tend to etch faster than surfaces with more copper, and once you are more experienced, you may want to use a stronger concentration for surfaces which require a lot of etching. The copper corrosion normally starts from the edge of the board and works its way to the center. Be sure to keep on agitating the board so that the resultant copper chloride (a powdery precipitate black in color) will get swept off the surface. This will speed up the etching process.

While it is important to make sure every part of the board is sufficiently etched, DO NOT keep the board in the solution longer than absolutely necessary. This is because extended exposure will allow the etchant to get under the resist and affect the fringe of your tracks, resulting in ugly patterns. Experience will soon tell you how long to leave it in for the concentration you use. Normally everything should be done in 25 minutes but it may be less, depending on the size of the board, exposed surface, and the concentration of the solution. Proper timing is especially important when very thin running tracks are involved.

If you are doing double sided boards, you should at some point, turn the board over. In this case, unless you have special holders, you should not over agitate the tray as the copper chloride precipitate which sinks to the bottom of the tray is rather abrasive and may scratch off some of the resist on the bottom side. Other than that, the procedure remains the same.

Some of the shops sell special etching tanks which stand vertically and has a little electric motor to automatically agitate the tray. This is not suitable for small scale productions as the tank normally requires large amounts of etching solution to fill up, and cost quite a lot to buy. For me, the above method is more than sufficient.

## Drilling of Holes

The drilling of holes is typically the last stage of the PCB fabrication process so this may seem a little anachronistic. Nevertheless, this is the last common step of the various methods of PCB fabrication so I thought it'd be good to cover it now.

Clearly, you will need to drill the holes yourself if you intend to put components on the board. In some surface-mounted designs, especially common with microwave and UHF circuits, this may not be necessary. Unfortunately, you cannot use your trusty Black & Decker power drill for this purpose because of the excessive speed of the drill and the oversized drill bit. A hobby or hand-drill is suitable and cheap ones, both battery powered and mains powered, can easily be found in Singapore for under S\$50. You will need to get a few common small sized drill bits for PCB use. The most useful by far is the 0.8 mm drill bit. The 1 mm and 2 mm drill bits also come in handy when drilling larger holes on the PCB. Generally drilling PCBs do not require a lot of effort because the PCB material is relatively soft and easy to drill. Be sure to get spare bits because the bits tend to break easily and are rather brittle due to their small cross-sectional area.

You should position the drill bit perpendicularly to the PCB for drilling any holes, and always maintain a steady and firm grip of the drill. If necessary, you may want to use a sharp instrument to slightly indent the spot you want to drill, as a guide as sometimes the drill bit tends to spin away from the point and scar the rest of the copper surface. Usually though, properly made boards should have these guides etched in. Do not apply undue force as this might cause the bit to break or the board to crack. Apply a steady force on the drill until you feel the penetration of the PCB. It is also advisable to have a piece of unwanted even wood surface beneath the board so that you won't destroy your workbench or your drill bit. Soft-wood is best but other soft material will also do, eg. old hard cover books.

Normally the drilling process produces a substantial amount of debris which will obscure your drilling template. Thus you will want to drill holes systematically so as not to miss any holes inadvertently, and to drill a section at a time, clearing away the debris as they accumulate. Do not have the fan blowing while you are doing this or your XYL will be all over you for messing up the place! Once you have drilled all the holes, inspect the board for undrilled or partially-drilled holes. Also be on the lookout for tracks that may have come off as a result of the drilling. This may sometimes be the case when drilling large holes on a small pad. Remove burrs from the holes and then your board is ready.



## Masking the PCB

As was mentioned in the basic theory section, there must be a way of controlling which parts get etched and which parts of the board don't. I also briefly mentioned a number of methods. Here I will highlight two of the methods most relevant to us homebrewers. Direct penning onto the board using etch resist pens and photo-resist.

### Using Etch-Resist Pens

You can actually draw the desired tracks or patterns onto the copper clad board with etch resist ink. Get a normal copper clad board that has been cut down to size, washed and dried completely. Do not soak the board in the water for too long or the water may damage the board. Be certain to make sure that there is not grease on the board or oxidized surface. If necessary clean the board with some mild abrasive to obtain a shiny surface. Avoid touching this surface with you fingers or dirtying it. This will ensure a more even etching later on. There is no need to specially buy etch-resist pens for this purpose though you could do so. For simple purposes, permanent markers or Indian ink seems sufficient for the job. There are advantages and disadvantages of using such a method. On the plus side, this is a very convenient method for producing one-off, not too intricate or complex patterns, and can be done rather quickly. However, you cannot obtain high resolutions tracks or any degree of evenness with this method. The results tend to look amateurish. Just as a reminder, the tip of the etch-resist pen tends to dry up quite quickly so the pen should be re-capped tightly when not in use. Have a piece of paper near by to get the ink flow even before tryin to mark the PCB with the pen.

Sometimes you can buy rub-on transfers for tracks or pads which you can incorporate as part of your pattern to make it look neater. On the whole, however, this method is reserved mostly for experimentation or very simple circuits with broadly spaced tracks. Alternatively, you can also use special tracking adhesives to paste out your tracks. Either way, the end result is rather coarse and difficult to reproduce.

Recently, there are available in the United States, special transparencies which you can laser print or photostat your track onto, and then iron-off the pattern from the transparency onto the board. Below is an excerpt which says something of this method:-

From: gary%ke4zv.UUCP@mathcs.emory.edu (Gary Coffman)

There is a special transparency film called Tec 200 marketed for this purpose, but I've found that Avery overhead transparency film works just as well, and is available at most larger computer or office supply stores. You just print your board layout to the transparency with your CAD package laser driver, remember you want a mirror image, and then iron it onto the copper. The copper needs to be clean, just as it would be for any resist application. You need a fairly hot clothes iron to fuse the toner to the copper. I use a regular home iron set for "cotton" and use an old Tee shirt between the iron and the film. After it cools, you can peel the transparency film off the circuit board and the toner will remain behind as the resist pattern. There may be a few pinholes or gaps where the toner didn't transfer well. You can patch them up by hand with an ordinary resist pen.

Note you can also use Avery film in ordinary copiers to generate a transfer from magazine artwork or hand drawn paper layouts. Of course when laser printing the film, you need to adjust your CAD driver so that the laser printer gives a properly dimensioned copy, and when using a copier, one with infinitely adjustable "zoom" feature is handy for the same purpose. If the artwork is "normal", you can first make a copy to a transparency, flip it over, and use that as your master for making the transfer transparency.

Works good, costs little.

Another method I have come across of directly masking the PCB is through the use of flatbed plotters. Apparently, the ink used in these plotters are etch-resistant and if you can design the board using CAD software, you should be able to plot the mask directly onto the board using the plotter. I have not tried this myself but a friend of mine has and reports good success.

#### Photo Resist Masking

This is probably the best way I know for making nice looking PCBs. Unfortunately, the technology behind it is rather obscure for many people entering the hobby and remains a mystery for others. Thus I will try to demystify the process here, with some luck. Contrary to the belief of many, the photo-resist method does NOT produce tracks on the PCB, it only produces a mask or pattern of etch-resist material, after which the board still needs to be etched like in all the other methods.

In this method, you need to get your pattern or mask onto a piece of clear transparency. This is usually done by laser printing direct on to the transparency, or photostating on to it. This means that anything that can be photostated, eg. patterns from magazines or from the ARRL handbook, or even texts and pictures, can be etched. This adds a number of advantages. For one, it is much easier to draw patterns on normal paper than on the copper surface. There is no need to use special etch resist ink for this purpose. You can also draw lines with higher density and definition as well as accuracy than you can using the direct method. You can use PCB layout software to print out computer generated patterns as well as including printed texts as part of the pattern. The possibilities are numerous. It should be noted that all the patterns must be black and white, no grays, and that the transparency must be clear, clean and colorless. Transparencies used for OHP presentations are suitable for this purpose. As an additional hint, you should try to get the transparency prepared such that the side with the toner is also the side that will be in contact with the PCB during exposure. This yields slightly better defined lines as there is then only one clear edge. It does not matter that the print is not completely opaque when you look at it against the light, usually normal photostat contrast is sufficient. You may want to cut the transparency to the size of the PCB for easier handling. Do not scratch the transparency as the toner may come off. If you notice missing tracks, you can still fix it by drawing on the missing tracks using an opaque black marker pen. If you notice excess tracks, slowly scrape off the toner/ink gently using a paper cutting blade. One advantage is that once you have produced one mask, you can use the same mask to produce a number of identical boards. When producing the mask, you should try to get it so that the emulsion side (the print side) is the side that contacts the PCB. This way when you expose the board, there is a minimum of shadow and fringe effect at the edges of the tracks and results in higher definition tracks.

The copper clad board must be specially prepared or sensitized by spraying a film of photo-sensitive masking material on to it. This spray is normally available in a canister and leaves a coat of clear green color (usually) when applied to the board. Spraying must be even and a sufficiently thick film must be deposited and dried before commencing exposure. All this should be done in low light/UV conditions as the spray is photo-sensitive. Alternatively, and more conveniently, boards that have been presensitized can be purchased quite easily from the shops. In any case, the spray is very expensive and not easy to use. These pre-sensitized boards come in light-proof wrapping which you may remove. The boards have a second protective plastic film over the

surface so you need not worry about accidentally exposing the boards. The rate of reaction is way slower than that of the camera film so you need not be overly concerned of over exposure. Just be sure that you are not doing this under intense fluorescent or sunlight. The second protective layer is an opaque adhesive plastic layer which is stuck to the board surface. This is usually white in color. Do not peel off this layer until you are ready to expose the board. If you do accidentally peel it off pre-maturely, store the board in a dark place until you are ready. In any case, these boards need to be stored in the dark and in a cool environment.

Once your transparency is ready and you have cut the board to size (without removing the protective layer), prepare yourself a clear piece of flat glass such as that found in picture frame. Be sure that the glass surface is clear and clean, and that its size exceeds the size of the PCB. This glass is used to hold the transparency to the PCB during exposure. Put the PCB on a flat surface and align the transparency over it, making sure that when you look at the transparency, you see the exact image of the track/pattern that you want, NOT the mirror image NOR the negative. Be sure your UV source is NOT active. Once you are ready and have double-checked every detail, slowly peel off the protective layer from the pre-sensitized PCB and replace it on the flat surface. Under the protective coating you should see a hard and dry, green film over the copper. Place the transparency correctly over the PCB and align it. Then, place the piece of glass over the transparency to press it firmly to the PCB surface. Once again check your alignment and then expose the board to the UV source.

The UV source can be a table top fluorescent lamp, or the sun, or special UV lamps. In all cases the UV content is not the same, thus exposure time varies. In my case, I use a table top lamp with an 11-watt fluorescent tube and place it about 2 to 3 inches above the board for 6 minutes to give me a properly exposed board. Under the afternoon sun on a clear day, it takes about 8 to 15 minutes to get sufficient exposure. Under UV lamps, the period may be as short as 30 to 90 seconds depending on the intensity of your source. Experimentation is the key to knowing how long to get the right exposure. Excessive exposure will damage the board and under-exposure will be equally disastrous. Once you have determined the correct exposure time, however, it is the same every time when using the same type of board, so be prepared to experiment a little with your first few boards. NEVER move or adjust the board once you have started exposure. Once you get good at it, you can even expose a number of boards simultaneously. Some types of board will exhibit a slight color change on the exposed parts once they are done but do not count of this method to determine when to end because the change is barely

perceptible. Note that if you are using a UV lamp, be careful not to look at the light direct as it may damage the eyes because the iris of the eyes do not respond too well to UV and may result in retinal-burn.

During the few minutes of exposure, get the developing solution ready. This solution is normally sold in the same shops where you purchased the PCB in the first place. They sometimes come under the name of POSITIV 20 or something similar and consists of an alkaline solution. Have this ready when you finish exposing. If you observe the board carefully, you may notice that the exposed portions are a little lighter green in color than the masked portions. This allows you to actually see a faint trace of your masking pattern on the exposed board. Rinse the exposed board in the developer solution and if properly exposed, you will see the exposed parts of the green photo-sensitive film dissolve in the developer solution. Once the unwanted parts have been completely dissolved and washed away, rinse the board under cold running water to remove any remaining developer solution. You should now see a very clearly defined, green, image of your original pattern on the PCB now. Dry the board carefully, making sure that you do not accidentally scratch off the resist/film. At this point you can still make corrections to the pattern using etch-resist pens or by scraping off resist/film from excess sections. Once everything has been confirmed, put the board aside and prepare for etching as outlined above.

#### Making your own PCB layout masks

There are a number of ways you can use to produce your own photo-exposure masks and layouts. Typically you want to draft out the layout on paper first before committing it to the final mask. Be sure to take into consideration RF paths and good grounding. There are a lot of considerations that need to be taken into account of in the design of a good PCB layout. Once you have drafted out the layout you can use hand-drawn masks, or combine hand-drawing with the use of Decal-Dry or rub-on transfers. These methods are suitable only for low density/complexity designs. The easiest way however is by the use of CAD software. There are some easy to use but fairly competent PCB CAD shareware available and if you intend to produce PCB designs of your own, you should be familiar with such software. Describing how they work is outside the scope of this article but among the features of such software, are their flexibility, multiple printer support, multiple layer support, silk-screening support, automatic drill guides on pads, auto-routing, easy editing, free/shareware, standard component templates and the list goes on and on.

## Summary

In summary, let me outline the steps and tools involved in the direct PCB fabrication method. First, the tools and materials:-

- a) Ferric Chloride or Sodium Persulphate crystals (or solution).
- b) A plastic tray big enough to immerse the board fully.
- c) The single or double sided copper clad board.
- d) Etch resist pen and/or transfers.
- e) A small medium speed drill with 0.8 mm bits.
- f) Hobby saw or Exacto knife to cut the PCB down to size.
- g) File to give the board a good finish.
- h) Mild abrasive for removing the resist from the PCB after etching.

The steps involved are as follows:-

- a) Prepare a draft of the desired layout.
- b) Cut out the required size of the copper clad board.
- c) File the edges of the cut down board for a smooth finish.
- d) Transfer the layout to the copper clad board by drawing it on with the etch resist pen or transfers.
- e) Double check for errors.
- f) Prepare the etching solution as by adding 1 part crystals to 4 or 5 parts water. Refer to section on etching.
- g) Immerse the masked board into the tray with the etching solution.
- h) Agitate the tray slightly for about 15 to 25 minutes, paying attention to the extent of the etch.
- i) Remove board from tray when completely etched.
- j) Rinse board under cold running water from the tap.
- k) Dilute used etching solution with lots of water before disposal.
- l) Use the mild abrasive to remove the etch-resist from the board.
- m) Use the drill to drill the appropriate holes for the components.
- n) Remove burrs from the holes.

For the photo-resist method, the tools required are the following:-

- a) Ferric Chloride or Sodium Persulphate crystals (or solution).
- b) A plastic tray big enough to immerse the board fully.

- c) Pre-sensitized copper-clad board.
- d) Transparency suitable for photostating.
- e) UV light source.
- f) Developer solution.
- g) A piece of clear glass to hold mask in place.
- h) Marker pen or transfers.
- i) A small medium speed drill with 0.8 mm bits.
- j) Hobby saw or Exacto knife to cut the PCB down to size.
- k) File to give the board a good finish.
- l) Mild abrasive for removing the resist from the PCB after etching.

The steps involved in the photo-resist method are as follows:-

- a) Prepare the masking pattern on a piece of white paper.
- b) Transfer pattern to the transparency by photostating.
- c) Cut the pre-sensitized board down to size.
- d) File the edges to remove unevenness.
- e) Place transparency on the board to check alignment.
- f) Peel off protective layer from board.
- g) Align the transparency on the board.
- h) Place glass over the transparency to hold it firmly in place.
- i) Place the UV source over the board and glass.
- j) Activate the source and expose board for a suitable period. Read above.
- k) Rinse the exposed board with the developer solution to dissolve unwanted resist.
- l) Double check for errors.
- m) Prepare the etching solution as by adding 1 part crystals to 4 or 5 parts water. Refer to section on etching.
- n) Immerse the masked board into the tray with the etching solution.
- o) Agitate the tray slightly for about 15 to 25 minutes, paying attention to the extent of the etch.
- p) Remove board from tray when completely etched.
- q) Rinse board under cold running water from the tap.
- r) Dilute used etching solution with lots of water before disposal.
- s) Use the mild abrasive to remove the etch-resist from the board.
- t) Use the drill to drill the appropriate holes for the components.
- u) Remove burrs from the holes.

## Conclusion

The real key to learning to make PCBs is to do it yourself. In this article I have tried to provide a general idea of the process of fabricating your own PCBs and have purposely included a number of cautionary and warning notes so that the reader will be aware of the hazards involved. On the other hand I have been

making my own PCBs for about 8 years now and have not suffered any side-effects or harm. Hopefully, this article will open new doors and possibilities for the homebrewer and that through homebrewing, one very significant aspect of the original spirit of Amateur Radio may be restored. If there should be further inquiries, I will be more than glad to help out.

73,  
Daniel Wee

--

```
+-----+-----+
| Daniel Wee | daniel@pandora.lugs.po.my |
| 9V1ZV      | daniel.wee@f516.n600.z6.fidonet.org |
| UUCP1.12j  | Packet: 9V1ZV @ 9V1VS.SGP.AS -- |
+-----+-----+
```

From owner-qrp-l@netcom.com Fri Mar 17 18:50:33 1995  
Message-Id: <n1416673558.25336@msmailgw1.arlut.utexas.edu>  
Date: 17 Mar 1995 14:00:49 -0600  
From: "rohre" <rohre@msmailgw1.arlut.utexas.edu>  
Subject: Printed Circuit Layout Supplies

Dan, there are a number of ways to go. I think your email is from Buffalo, and in a city that large, there are probably several Drafting Supply companies for industrial drawing supplies. QST among others has advertised for someone who makes a film that is exposed to a layout pattern in a Xerox, and the film is then applied to a PC board. I don't remember all the details, but this forms the negative for exposing a sensitized board to light, and thus creating which areas are to etch. The drafting supply companies or industrial electronic distributors should have some materials.

Another product I have seen at industrial electronics parts houses is from Bishop, a Ca. company, I think, who made copper stick on pads, copper tape, socket patterns, etc. so you took a fiberglass board and put the copper over mylar tape right on it without any etching chemicals or mess. You solder over the tape to pad junctions, etc. That is about the same as using the black drafting supply co. layout tape and donut pads to make the Master negative, but you save the photo and etch work. For one shot projects it is quite acceptable. An Xacto hobby knife helps you push the tape where you want it, and get it stuck tightly to the board, for either method.

There are of course non printed circuit methods for simple projects and even the complex, where you modularize the circuits onto terminal equipped insulating boards, and use point to point soldering. These have been available in both the U. S. and in the Commonwealth countries, RS Components (Radiospares U.K.) had these tag boards some 30 years ago when I built my first SSB qrp rig.



Hope that points you toward a solution.  
73 and 72, Stuart K5KVH

From owner-qrp-l@netcom.com Fri Mar 17 23:21:48 1995  
Date: Fri, 17 Mar 95 22:06:15 -0500  
Message-Id: <9503180306.AA08187@host.igs.net>  
From: ve3lfo@igs.net (Dave Burniston)  
Subject: QRPp in Canada?

Any Canadians received their QRPp yet? Still no sign of mine.

Dave  
VE3LFO

From owner-qrp-l@netcom.com Fri Mar 17 22:27:56 1995  
Date: Fri, 17 Mar 1995 18:22:29 -0800  
From: faunt@netcom.com (Doug Faunt N6TQS 510-655-8604)  
Message-Id: <199503180222.SAA20493@netcom8.netcom.com>  
Subject: Re: RFC value...HOW??

Useful info on radio frequency chokes (how many others think Request  
For Comments when they see RFC?), now how about info on using ferrite  
beads instead of RF chokes?  
73, doug

From owner-qrp-l@netcom.com Sat Mar 18 01:31:48 1995  
From: JimN0OCT@aol.com  
Date: Fri, 17 Mar 1995 23:59:52 -0500  
Message-Id: <950317235951\_53120256@aol.com>  
Subject: RS12/13 ops (long) was:AK QS0...

Hi y'all,

Am copying the list on this FYI, if anyone is interested in 'bird work'.  
-----

Hi Ron,  
You wrote:

-----  
OK Jim,

Been playing with these here radios for over 40 years  
and ain't never done molested a satellite before. So  
tell me more.

Freq's and what do you do? Must be some sort of "rules"

to play by. I can do 10 and 15.

Geeeeesssh, more toys to play with now.....

73's, Ron

.....KU7Y.....Monte "Ron" Stark.....  
....ku7y@sage.dri.edu.....Sun Valley, Nevada....  
.....ARRL.....NorCal #330.....NRA LIFE.....  
-----

Well, to start with, RS-12/13 is a low earth orbiting (LEO) satellite. It's frequencies of operation are 21.210-21.250 MHz uplink, 29.410-29.450 MHz downlink. So what you need is a transmitter and receiver with separate antennas (dipoles are fine). This is full duplex work, like the phone, so you can hear yourself talk. Normal use requires Advanced or higher privileges; Novice work and higher is possible on 21.129/29.454.

If you have a transceiver with separate receive and transmit antennas, and dual VFO's you're set. I don't, so I use my transmitter on 15 meters, and an R390/A for a receiver (with its own antenna). Dipoles work great for this (for both rigs), and I have even completed QSO's with a wire dipole in the basement on the receive side. You will also need to use headphones to prevent feedback on SSB, not entirely necessary on CW.

Some power will help. I haven't run RS 12/13 since I've been QRP only--used to use my TS-820 S for transmit. With a QRO rig, it is important to "ride the output"--in other words, use only enough power to hear your self on the downlink (I never needed to turn it up all the way to 100 Watts). Too much power up taxes the satellite and can be detrimental to it.

You'll need either a satellite tracking program for your computer, or a friend who has one and can print out the pass times for you. You will also need a recent copy of the two line element sets (elsets) for the bird. ARRL lists these in some of their bulletins, and there are ftp sites and BBS that carry this info. These will need to be entered into the tracking program. Good programs to use are: PCTRACK, Traksat, and some others I'm not familiar with. Look for one that allows you to print out pass times (I think PC track does). I like to have a printout to refer to while doing this.

You will need some manual dexterity. More on that later.

To start with, once you have the pass times in front of you, choose pass times when the band will be 'dead' or 'unused'. By dead I mean 15 meters at the bottom of the sunspot cycle at night. By unused, I mean a weekday during the Sunspot cycle minimum. The reason for this is that being an HF band, 15 meters may have some activity on it that A: won't be working the satellite, but will be received and downlinked by the bird causing QRM for you and

B:will hear you and try to work you, but you won't hear them unless they are passed through the satellite! Everyone who has worked RS 12/13 has had their share of 'busted' QSO's (ones that don't count as satellite QSO's) this way. Some guy/gal comes back to you, you get excited, then the other op asks you why you were calling CQ RS 12/13 and mentions how well you're getting into Freemville on 15 meters. So, operating during the evening is a good practice for beginners.

The first thing to do is start listening for the satellite's beacon on 29.408 MHz about 5 minutes before your scheduled AOS (Acquisition Of Signal). Once you hear the beacon fairly strongly, tune the receiver up into the passband of the downlink. Transmit a few dots with the transmitter (asking first [simplex] if that frequency is clear) until you hear your signal in the receiver. At this point, you can call CQ. On CW, most people call CQ RS or CQ RS 12. On voice, they call CQ Satellite or CQ RS12. In general, the lower end of 15 is used for CW, and the upper end for SSB.

One thing about satellites: because of the Doppler effect, as the satellite goes overhead your received frequency will change. Some people change the transmit frequency to stay in their receiver passband; I change the BFO on the receiver to track my frequency so that my transmit frequency is constant. If i'm in QSO with someone, the other station won't have to try to track my signal and his at the same time--we both would just keep the our respective receivers tuned to the downlink signals. This is where manual dexterity helps.

On a directly overhead pass, you will have about 15-18 minutes of operation. Because of this, there are usually two kinds of responses to CQs: ragchews, and "work-em-alls". For 'work-em-all' QSO's, exchange your information as quickly as possible and move to the next QSO. You try to work as many stations in one pass as you can. I've had 4-5 QSO's in a pass; a good contest op will get more.

Chances are you won't "work the world" with RS-12/13. Because it is a LEO satellite, its "footprint" (the area of the earth that is line-of-site to it at any given time) is fairly small, and if I remember correctly it is about 4500 mi in diameter. So, from St. Louis I've got a fairly good chance of working all the 48 continental states and even Alaska. Remembering that the uplink and downlink are HF freq.'s, however, does allow for some interesting propagation. People have worked DX on RS 12/13. If you're interested in trying this, leave your receiver tuned to the beacon frequency all the time--you may hear it when it is not overhead! If so, give a shout and try to work it.

Whew! Sorry for the bandwidth, but RS 12/13 is a fun satellite almost anyone can operate. No special 2 meter equipment required. Many of the computer tracking programs are shareware (not freeware) and work well. If anyone has any other questions (or would like to set up a sched) email me. QST also ran

a very good article in the Feb '94 issue (pp. 58-60). Good luck!

73 ES CQ RS12 CQ RS12 de N00CT

From owner-qrp-l@netcom.com Fri Mar 17 20:33:00 1995  
From: PDouglas12@aol.com  
Date: Fri, 17 Mar 1995 16:58:01 -0500  
Message-Id: <950317165621\_52749963@aol.com>  
Subject: signing /QRP

There's a very nice QRP primer in April's CQ Mag by Rich Arland, K7YHA. He raises an interesting point that may be a good topic for discussion here. He says some QRPers sign /QRP and some don't. Some think it helps, and some don't. He concludes "it wouldn't hoit" (my language, not his).

I don't agree entirely. On 7.040 maybe it helps us find our brethren.

Calling CQ is usually such a waste of time with low power, maybe the /QRP helps. But not with QRO stations, in my experience. Calling just plain CQ in the Extra portion, for instance, is sometimes fruitful, as it is a little less populated, and there are presumably better ops there with more experienced ears (I can hear the scornful laughter in some quarters already.)

But most QRO stations hear a weak sig and hesitate to answer because they don't want to struggle for a QSO. When they hear the /QRP suffix this just confirms their suspicion that this QSO will be a struggle (read pain in the ass). And the QRO guys already have a philosophical aversion to working so hard for their contacts, because challenge isn't as big a part of their hobby. So when I call CQ in the Extra end, or even just away from the QRP hangouts, I leave it off. On the other hand, I often add it after I have snagged some unsuspecting QRO guy into a QSO. That can really be fun, especially if he has already given me a 599+ (and he's running a real full gallon). I once embarrassed a Copenhagen kilowatter by working him 59 with a bare Argo 509 in the car (ten meters, height of the suncycle--but it counts). We all have stories like that, I bet. Signing QRP will automatically reduce the S meter on the QRO guy's S-meter, if he knows before giving the report. (I know, everybody already knows I'm a cynic. Save your flames please, I not worthy.) Anyway, my (wonderful) Idiom Press programmable keyer has a plain CQ in the first memory, and a /QRP CQ in the second.

Neither of them generates pileups with 5watts!

Finally, if I have revived an old subject that predates my belonging here, forgive me, but I haven't seen this discussed in the last six months at least. On an unrelated subject, I will upload the Dayton or Bust (long) list latest update late tonight. (You all got the alliteration?)

Preston WJ2V

From owner-qrp-l@netcom.com Fri Mar 17 21:41:38 1995  
From: N5EM@aol.com  
Date: Fri, 17 Mar 1995 18:42:43 -0500  
Message-Id: <950317184231\_52848986@aol.com>

Subject: Re: signing /QRP

Well, my experience with signing QRP is different. I have found that in listening to the calling frequency, I am interested in working fellow QRP stations. No real prejudice here, I just am monitoring the calling frequency to work QRPers. If I wanted to work anyone, I'd not pick the national 40 meter tune-up/errant amtor frequency.

So, I appreciate it when a fellow QRPer lets me know that he/she is qrp by appending QRP (Note: NOT /QRP) to the call sign. When I call CQ, its usually in the format, "CQ CQ CQ de N5EM N5EM N5EM QRP". It was discussed a number of years ago that signing stroke "/" anything implies something FCC related. Not to labor this point, though. I doubt anyone ever got a pink slip (they really are pink, but that's another story) for signing /QRP.

If you like the bit of deception of waiting for the 599 before telling about your station, go ahead. Just realize that if the band is good, and you don't say you are QRP, I may hear you and not call you because your signal is too good and I'm looking for a nice 2 way QRP qso. I am frequently called by QRO stations who are curious about the QRP add on. I am always delighted to chat with a QRO station, I just don't go out looking for them on the QRP calling frequency.

So, the moral of the story is, do whatever you like. But, don't hesitate to sign that you are QRP. You never know. I might be lurking just waiting for you to stop by.

And, not that this word is so definitive that I won't respond to your reply, I'll be skiing in Utah next week so I will not be checking mail. Some things take precedence to ham radio (for at least a while!)

72/73

Ed Manuel - N5EM  
n5em@aol.com  
Houston, Texas

QRP - Its a state of mind, not just a power level.

QRP ARCI #4914, G-QRP #1243, NORCAL #498  
(And, waiting on my CQC number!)

From owner-qrp-l@netcom.com Sat Mar 18 03:54:05 1995  
Date: Sat, 18 Mar 1995 01:03:47 -0500  
Message-Id: <199503180603.AA25997@yfn2.ysu.edu>  
From: ah301@yfn.ysu.edu (Jerry Sy)

Subject: Re: signing /QRP

>counts). We all have stories like that, I bet. Signing QRP will  
>automatically reduce the S meter on the QRO guy's S-meter, if he knows before  
>giving the report. (I know, everybody already knows I'm a cynic. Save your f

I completely agree with the above statement. I have been doing cw for  
2.5 months only, and most qso's with qro stations, as soon as I  
give my rig/power , they will qrt! so I usually don't give my  
rig description.

I only got one response so far since I started cw if I call CQ, which  
is my very first CW contact! never since after that has anybody  
come back to my CQ, whether I have /qrp or not. But I have 95% success  
if I answer a CQ.

73 de jerry  
AA3KN

p.s. Preston, pls change my call from n3rkd to the new one in your  
dayton list. tnx.

From owner-qrp-1@netcom.com Sat Mar 18 03:58:08 1995  
Date: Sat, 18 Mar 1995 02:00:36 -0500  
Message-Id: <199503180700.AA29046@yfn2.ysu.edu>  
From: ah301@yfn.ysu.edu (Jerry Sy)  
Subject: Re: signing /QRP

hello again.

just right after I posted a reply to this subject,  
I heard a CQ calling on 7.030, so I answered,  
the station went through his routine, then I wen through mine.  
He gave me a 599 report in Clearwater FL, I also gave him  
a 599 here in Pittsburgh PA. Then he says his rig is  
running 500 watts to an antenna on the roof of a 15 storey  
bldg. Then I told him I was running 5 watts qrp!  
Right after that, he said 73 got to QRT, and I was not  
even finished doing my finals 73s/CULs, he was already in  
QSO with another station!!

boy, talk about rough.

73 de jerry  
AA3KN

From owner-qrp-l@netcom.com Fri Mar 17 19:09:37 1995  
From: penc@psuh02meteo.psu.edu  
Date: Fri, 17 Mar 1995 22:44:36 GMT  
Message-Id: <95031722443628@psuh02meteo.psu.edu>  
Subject: Solder flux remover- the end

Gang:

Thanks for the responses to the query on removing solder flux from an old PC board. I received many many replies. Thanks to all. BTW, the isopropyl alcohol did the trick nicely, but I'll keep the Radio Shack flux remover in mind if I ever want to do it the right way.

Thanks agn and back to QRP.....

Rich WK2A  
listening

From owner-qrp-l@netcom.com Fri Mar 17 21:39:47 1995  
From: Byron8LCZ@aol.com  
Date: Fri, 17 Mar 1995 18:06:33 -0500  
Message-Id: <950317180609\_52815189@aol.com>  
Subject: Ten Tec Xmitting Attenuator Pad

Does anyone own the Ten Tec Transmitting attenuator pad ? If so, how about a description of it, size, weight, cost and number of steps and if a means of measurement is provided. Is it a simple PI or T attenuator like the handbook shows for receivers with higher wattage resistors ? Does it use slide switches ? I have never seen a picture of one. What are the steps 6 db ?

72, Byron WA8LCZ

From owner-qrp-l@netcom.com Fri Mar 17 19:52:30 1995  
Message-Id: <n1416683815.8599@msmailgw1.arlut.utexas.edu>  
Date: 17 Mar 1995 11:09:55 -0600  
From: "rohre" <rohre@msmailgw1.arlut.utexas.edu>  
Subject: What makes a Radio Frequency Choke

John,  
This is a good question, and some of the values you see in homebuilts are not optimum for the application.

The choke must oppose the flow of RF with a high impedance over the frequency range in which the rig operates. That is say, 10 times the impedance of other paths, such as the 50 ohm antenna. Some say you can go as low as 4 times the impedance. The point is it must make the RF want to go elsewhere.

Now the formula from the amateur radio handbooks for reactance which is one part of impedance, is for inductors:  $X_L = 2\pi F \times L$ , where  $2\pi$  is of course 6.28, and  $F$  is the frequency, and  $L$  is the inductance.  $X_L$  will go toward zero as the series circuit consisting of the choke reaches series resonance, and thus lowest impedance. To keep that from happening, you must build the choke to resonate above the frequency where it is to be used. Below its resonance, it will appear as an inductor and have inductive reactance to block the RF. At resonance it will cease to function as a choke. Thus in addition to selecting wire to carry the DC current and RF current, you must have a suitable value of inductance to keep the series resonance above the frequency band in which you want to block the RF from going say to the power supply instead of to the output tuned circuit of a transmitter. And I might add, an inductance to keep the inductive reactance higher by 4 to 10 times than the other preferred path for the RF to take; the one you want it to take. Hope this helps, 73, Stuart K5KVH